CFD SIMULATION STUDY ON THE EFFECT OF LEAK SIZE TOWARDS OIL LEAKAGE FROM SUBMARINE PIPELINES

MAHADI BIN BAHARI

Thesis submitted in partial fulfilment of the requirements
for the award of the degree of
Bachelor of Chemical Engineering (Gas Technology)

Faculty of Chemical & Natural Resources Engineering
UNIVERSITI MALAYSIA PAHANG

JULY 2014

ABSTRACT

This paper presents Computational fluid dynamic (CFD) studies on leak size effect towards the time taken for migration of oil droplets to reach free surface upon leakage. The size of the leakage hole (d) is a variable ranging from 0.1 m to 0.3 m with increment of 0.1m whereas the other parameter is fixed. A computational rectangular domain with length of 20m and height of 15m was simulated in Gambit 2.4. The length of computational domain is large enough, which is larger than the longest horizontal distance the oil droplets migrate when they reach the sea surface. In the Gambit 2.4, the time taken for the oil droplets to reach free surface was observed by varying the leak size; d1=0.1m, d2= 0.2m and d3=0.3m respectively. . The outer diameter (D) of damaged submarine pipeline is 0.5 m. The mesh was generated and exported to Fluent. Using the formulas we can obtain when and where to see oil reaching the sea surface, and conduct rapid response the maximum horizontal migration distance of oil at certain time is predicted, and a forecasting model is proposed. These calculated results will provide useful guidance to place the oil containment boom. Results were observed at 1000 number of time steps (iterations) with a step size of 0.1s. For the leak size 0.1m, the time taken for the oil to reach the free surface is 64s. For leak size 0.2m, , the time taken for the oil to reach the free surface had decrease to 55s. On the other hand, for the large leak size 0.3m, time taken for the oil to reach the free surface had decreased to 52s. As the conclusion, the leak size of pipeline can affects the time taken for oil to reach free surface upon leakage in pipeline.

Key words: Oil Spill, CFD (computational fluid dynamic), leak size, GAMBIT, FLUENT

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATIONS

<u>Greek</u>

v_l	kinematic viscosity
µm	micrometre
m/s	Water velocity / oil leaking rate
m^3/s	Volume flux
wt %	moisture content
KJ/kg	Calorific Value
Kg/m3	oil density
Pa.s	Pascal .seconds
Diameters	(height, distance)
ρ	density
g	gravity

CFD	Computational Fluid Dynamics
RANS	(Reynolds-Averaged-Navier-Stokes) equations,
VOF	Volume of Fluid
FVM	(finite volume method)
MESH	2.4.6
FLUENT	6.3.26

1 INTRODUCTION

1.1 Background Study

Oil has become one of the most important energy we have. Oil mostly had been used for transportation and generates energy for supply the power to system. Therefore, the demand for this energy is quite large and increasing. This issue had led to the exploration of oil and then the construction of submarine pipeline system at the sea. The important issues related the submarine pipeline is the oil spill or leakage incident. These incidents usually present in pipelines due to several factors such as corrosion, flow erosion, or submarine landslide. This incident may lead to serious environmental issues especially to marine life and human health. As the largest accidental marine oil spills had occurred in Gulf of Mexico, around 4.9 million barrels of oil were released into the sea. Due to the months-long spill, along with adverse effects from the response and clean-up activities, extensive damage to marine and wildlife habitats, fishing and tourism industries, and human health problems have continued through 2014 (Tangley,2010).

When this incident occurs, a quick and adequate response must be required in order to minimize the consequences especially to marine environmental. Therefore the accurate information related to the rise process of oil droplets and the dispersal path of oil spill is required. Through the exact information the oil containment boom can be set up immediately to reduce the damage of oil spills. A lot of researched and studied had been made to focus on the oil spill. At present, the modeling for forecasting oil spill behavior and incidence is usually based on sea surface (Zhu and Dmitry, 2002; Xie et al., 2007) or offshore zones (Guo and Wang, 2009; Guo et al., 2009). However, the numerical modeling for submarine oil spill is relatively lacking. Some research has forecasted the trajectory of submarine oil spill using radar galvanic current (Abascal et al., 2009), but the approach can only supply partial real-time information and may not support emergency behavior for the influence of weather and night. Li and Yapa (2002), Øistein et al. (2003) and Dasanayaka and Yapa (2009) have also carried out the research on submarine oil ejecting, but they all aim at oil gas mixture and cannot contribute to forecasting oil spill greatly.

However, for oil leakage from damaged submarine pipeline, the trajectories of oil spill that flow along the depth direction are important issue need to be considered. An effective attempt has been made to observe the oil spill under the action of current and wave (Li W, 2013) .However, the velocity of current in their study was uniform, which does not match with the actual shear velocity distribution under sea surface and the actual hydrostatic pressure distribution was not used in their modelling. According to Yadav and Jalilinasrabady (2013), the information about hydrodynamics of oil flow, which are not easily obtained through the physical experiments can be obtained using numerical simulation. Therefore, in this researched, CFD (computational fluid dynamic) simulation with FLUENT software had been used to investigate the process of oil spill from submarine pipeline to free surface.

In this CFD simulation, the effect of oil leakage size is examined by using a several series of numerical simulation. Then, the dimensionless time required for oil droplets which have the longest horizontal migrate distance when they reach the sea surface and the dimensionless longest horizontal distance the droplets migrate when they reach the sea surface are analyzed and the fitting formulas are obtained. Using the formulas we can obtain when and where to see oil reaching the sea surface, and conduct rapid response. Finally, the maximum horizontal migration distance of oil at certain time is predicted, and a forecasting model is proposed. The results provide useful guidance to place the oil containment boom.

1.2 Problem Statement

The increasing oil spill accidents have led to much oil leaking into the sea and had causing extensively damage toward marine environment and human health. A lot of research had been made to modelling the forecasting oil spill behaviour. However, the numerical modelling for submarine oil spill is relatively lacking. For the oil leakage from damaged submarine pipeline, the migration of oil flow along the depth direction is more important issue.

Therefore the CFD (computational fluid dynamic) simulation with FLUENT software had been proposed to solve this problem. In this method, the dimensionless time required for oil droplets which have the longest horizontal migrate distance when they reach the sea surface and the dimensionless longest horizontal distance the droplets migrate when they reach the sea surface are can be analysed and the fitting formulas can be obtained.

Summary

The topic was scoped from addressing the problem in the petroleum industry, way by indentifying the problem of leakage in submarine pipelines. Then, an alternative solution using the Computational Fluid Dynamic (CFD) simulations with FLUENT software to detect leakage process of oil spill from submarine pipeline to free surface was implemented. Then, an alternative solution using the computational fluid dynamics(CFD) simulation had been introduced . CFD (computational fluid dynamic) model coupling with VOF (volume of fluid) method has been used to investigate the process of oil spill from submarine pipeline to free surface.

1.3 Objective

The main objective of this research is to study the time taken for oil droplets(s) to migrate along a horizontal distance up to free surface with varying leak size of submarine pipeline. In this research, CFD (computational fluid dynamic) simulation with FLUENT v6.3.26 software are carried out in order to investigate the migration process and the trajectory of oil spill from submarine pipeline to free surface of sea.

1.4 Scope of this research

The scopes of this study are mainly to study the effects of oil leak size on the oil spill process. The method of study is by implementing computational fluid dynamics (CFD) using the Gambit 2.4.6 and the Fluent Software. The whole computational domain is a rectangle with a length of 20 m and a height of 15 m. The length of computational domain is large enough, which is larger than the longest horizontal distance the oil droplets migrate when they reach the sea surface. Water occupies the lower region with height of 14.5 m, while air occupies the upper region. The damaged of submarine pipe with the outer diameter (D) of 0.6 m. The size of the leakage hole (d) is a variable ranging from 0.1 m to 0.3 m with increment of 0.1m, in order to examine the effect of leak size. GAMBIT 2.4.6 mesh-generator is employed to perform all geometry generation and meshing. The scope also widens to compare the horizontal distance the droplets migrate in the given period of time. Finally, the leakage can be detected more accurately and cost-effectively.

1.5 Hypothesis

As the leak size (m) of submarine pipeline increases, the time taken for the oil droplets to reach free surface is much shorter.

1.6 Main Contribution of this work

The following is the contribution:

i) Contribution was prior to our supervisor's guidance in using this computational fluid dynamic (CFD)
ii) The leak size is the main factor that should be controlled during the oil spill. Therefore it is relevant to study about it in order to prevent more damage from oil spill.

1.7 Organisation of this thesis

The structure of the reminder of the thesis is outlined as follow:

Chapter 2 provides an overview of oil leakage in pipelines underneath the ocean. A general description of Computational Fluid Dynamics (CFD) and the Volume of Fluid (VOF) approached. This chapter also provides a brief review on previous study made on oil leakage myths. A comparison made on all the factors which directly affect the time period for the spills to reach the free surface.

Chapter 3 gives a review of the procedure involved in the simulation process. The computational domain and Mesh generator were simulated. Results were further generated for different cases comparing to its standard case.

Chapter 4 gives a clearer understanding of the effect of leak size on the length of time for oil to reach the sea-surface and the distance for oil moving downstream, simulations are conducted by changing the leak size while leaving other parameters same. With increasing leakage size, the time required for oil to reach the maximum horizontal migrate distance when it reaches the free surface is shortened. As leakage size reduces from 0.3 m to 0.1 m, the required time decreased will be decreased.

Chapter 5 draws together a summary of the thesis and outlines the future work which might be derived from the model developed in this work.

2.0 LITERATURE REVIEW

2.1 Screening Route

My literature study was on the oil leakage in submarine pipelines which endanger the environment and aquatic life. In addition, research was also completed on Computational Fluid Dynamics (CFD). The study conducted did not neglect the environmental aspects and economy imprecision. Finally, the literature centered on the effect of water velocity towards oil migration in a horizontal distance to the free surface.

2.2 Overview of Oil Spill Incident

An oil spill is the release of a liquid petroleum hydrocarbon into the environment, especially marine areas, due to human activity, and is a form of pollution. The term is usually applied to marine oil spills, where oil is released into the ocean or coastal waters, but spills may also occur on land. Oil spills may be due to releases of crude oil from tankers, offshore platforms, drilling rigs and submarine pipeline (Marybeth, 2004). The oil spill incident can caused numerous problems within the ocean ecosystem and has continued to cause problems even after some time has passed. Issues such as genetic damage, liver disease, and cancer can occur within the wildlife among other aquatic life defects. When the aquatic environment became toxic this affected humans too. As omnivorous consumers, fish are important to humans because of the important nutritious resources they provide. If these species become dangerous to eat, human health can be affected and may lead to abnormal disease or sickness.

2.3Factor of Oil Spill from submarine pipeline

There are several factors that may cause the oil spill from submarine pipeline. Table 2.1 below explain detail about the factor of oil spill incident occur.

Table 2.1 The factors of how oil spill can be occur from damaged submarine pipeline

Factor	Explanation	Author
Submarine landslide	This is happen due to high of sedimentation rates and usually occurs on steeper slopes. This landslide can be triggered by earthquakes in the sea. When the soil around the piping system is subjected to a slide, and give the result of displacement at high angle to the pipeline, the pipe will severe bending. This will cause tensile failure.	Palmer & King (2008)
Ice issues	This happen to submarine pipeline system in low temperature water especially in freezing waters. In this case, the floating ice features often drift into shallower water. Therefore their keel comes into contact with the seabed. When this condition happen, they will scoop the seabed and came hit the pipeline	Croasdale (2013)
	Stamukhi can also damage the submarine pipeline sytem. Stamukhi is a grounded accumulation of sea ice rubble that typically develops along the boundary between fast ice and the drifting pack ice. This stamukhi will exert high local stresses on the pipeline system to inducing the excessive bending.	Croasdale (2013)
Ship anchors	Ship anchors are a potential threat to submarine pipelines, especially near harbors. This anchor will give high damage to the pipeline due to their massive weight.	
Corrosion	For small size lines, additionally, failures due to external corrosion were more frequent compare than internal corrosion. However in medium and large-size lines, failures due to internal corrosion were more frequent than those due to external corrosion.	Mandke (1990)

2.4 Effect of Oil Spill from Submarine Pipeline

A serious environmental issue can be a rise from this oil spill incidents. It is very unlikely to occur, however if this incident happen, it would be likely to have serious consequences such as the incident in Gulf of Mexico. This consequences can be serious depends on many factors including the size of leak of oil spill, its location, the leak rate, and the weather. All this effects must be considered in the submarine pipeline spill consequence analysis. Table 2.2 below show the several effect of oil spill from damaged submarine pipeline (Iberahin, 1999)

Table 2.2 the effect of oil spill from damaged submarine pipeline (Iberahin, 1999)

No	Effect of Oil Spill
1.	The depth of submarine pipeline as referred to it location on the seabed will affect the rate at which the oil reaches the surface. The oil will become oil in water emulsion that will increase the persistence of the oil slick.
2.	An open sea activities such as fishing activities will be prevent because the cleanup operations are underway. Besides that, the equipment would be fouled and fishing would have to be temporarily suspended while oil slicks persist. The impact on fish stocks will normally be restricted to eggs and larvae.
3.	In the event of oil spill near to the shoreline the tourism, the recreational activities will be directly affected. Oil slicks would reach the beach and prevent activities such as swimming, sailing, diving, etc. In short oil pollution would disrupt water sport and cause acrimony among both tourists and local populace, with associated loss of tourist income.
4	When an oil slick from a large oil spill reaches the beach, the oil coats and clings to every rock and grain of sand. If the oil washes into coastal marshes, mangrove forests or other wetlands, fibrous plants and grasses absorb the oil, which can damage the plants and make the whole area unsuitable as wildlife habitat.
5.	The major effect of this oil spill is toward marine life. The oil sometimes clogs the blow holes of whales and dolphins, making it impossible for the animals to breathe properly and disrupting their ability to communicate. Besides that, the shrimp and oyster fisheries along the Louisiana coast were among the first casualties of the 2010 BP Deepwater Horizon offshore oil spill.

2.5 Previous Research

Most previous research and modelling only focused on the surface of two-dimensional oil spill. (Berry A, 2013; Gonzalez, 2008; Lamine, 2013). However it still lacks of the information how the migration of oil flows along the depth direction of sea. (Zhu, 2013)Therefore CFD (simulation fluid dynamic) with FLUENT software and VOF (volume of fluid) method had been used to examine the migration process of oil spill from submarine pipeline to free surface of sea. Table 2.3 below show the several researches that had been developed regarding the oil spill case.

Author	Year	Title	Remarks
Jen-Men Lo	1991	Oil-Spill Risk Simulation Model.	A simulation numerical model that can generate an oil-risk map for a given area. A model simulates a spill's location, size, and associated movement based on statistical data. Horizontal wind vector components are simulated using a Markovian time series model based on local wind statistics.
Xiaobo Chao, N. Jothi Shankar, Sam S. Y. Wang	2003	Development and Application of Oil Spill Model for Singapore Coastal Waters	Application of a three-dimensional oil spill model for predicting the movement and fate of an oil slick. In the model, the oil slick is divided into a number of small elements or grids for simulating of the oil processes of spreading, advection, turbulent diffusion, evaporation, dissolution, vertical dispersion, shoreline deposition and adsorption by sediment.
Vikram J. Kaku, Michel C. Boufadel, Albert D. Venosa	2006	Evaluation of Mixing Energy in Laboratory Flasks used for Dispersant Effectiveness Testing	The evaluation of dispersant effectiveness used for oil spills is commonly done using tests conducted in laboratory flasks. A hot wire anemometer to characterize the

			turbulence characteristics in the swirling flask (SF) and the baffled flask (BF).
Hao Xie, Poojitha D. Yapa	2006	Developing a Web-Based System for Large-Scale Environmental Hydraulics Problems with Applicaiton to Oil Spill Modeling	Used for decision making during emergencies, contingency planning, and risk assessment. Many of these oil spill models are desktop based. This could be useful include modeling sediment plumes and deepwater oil well blowouts.
Fanghui Chen, Poojitha D. Yapa	2007	Estimating the Oil Droplet Size Distributions in Deepwater Oil Spills	To estimate the oil droplet size distribution generated due to an accidental release. Models for estimating oil droplet size distribution generated by a deepwater release are developed based on the maximum entropy formalism.
E. Sarhadi Zadeh, K. Hejazi	2010	Modeling of Flow and Water Quality Processes of Oil Spills with Finite Volume Method	The movement of oil spill, its diffusion and hydro-environmental effects have been simulated by developing a 2DH numerical model based on non-linear shallow water Reynolds averaged Navier-Stokes equations using Finite Volume Method.
Hongjun Zhu, Pengzhi Lin, Qian Pan.	2013	CFD (computational fluid dynamic) simulation for oil leakage from damaged submarine pipeline	CFD (computational fluid dynamic) simulations with FLUENT software are carried out to investigate the process of oil spill from submarine pipeline to free surface. Effects of oil density, oil leaking rate, leak size and water velocity on the oil spill process are examined.

Table 2.3 the several researches that had been developed regarding the oil spill case.

2.6 Overview of CFD (Simulation Fluid Dynamic)

Computational Fluid Dynamics (CFD) provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of mathematical modeling (partial differential equations), software tools (solvers, pre- and post-processing utilities), and numerical methods (discretization and solution techniques). (*Wesseling* et al., 2001)

2.6.1 Experiments versus Simulations

Table 2.4: Comparison of experimental and simulation runs (*Wesselling et al., 2001*)

Experiments	**Simulations**
Quantitative description of flow phenomena using measurements • For one quantity at a time • At a limited number of points and time instants • For a laboratory-scale model • For a limited range of problems and operating conditions Error sources: measurement errors, flow disturbances by the probes	Quantitative prediction of flow phenomena using CFD software • For all desired quantities • With high resolution in space and time • For the actual flow domain • For virtually any problem and realistic operating conditions Error sources: modeling, discretization, iteration, implementation

2.6.2 The finite volume method.

A method for discretizing the transport equations commonly implemented in CFD codes are the finite volume method (FVM). In a FVM, the computational domain is divided in control volumes and conservation principles are applied to each control volume. This ensures conservation, both in each cell and globally in the domain, which is a great advantage of the FVM. Using FVM also allows for the use of unstructured grids which decreases the computational time. (Stenmark et al., 2013)

2.7 Multiphase flow theory

Multiphase flow is flow with simultaneous presence of different phases, where phase refers to solid, liquid or vapor state of matter. There are four main categories of multiphase flows; gas-liquid, gas-solid, liquid-solid and three-phase flows. ***(****Thome, 2004)*

2.7.1 VOF Model Approach

A third modeling approach is the volume of fluid (VOF) method. VOF belongs to the Euler-Euler framework where all phases are treated as continuous, but in contrary to the previous presented models the VOF model does not allow the phases to be Interpenetrating. The VOF method uses a phase indicator function, sometimes also called a colour function, to track the interface between two or more phases. The indicator function has value one or zero when a control volume is entirely filled with one of the phases and a value between one and zero if an interface is present in the control volume. Hence, the phase indicator function has the properties of volume fraction. The transport equations are solved for mixture properties without slip velocity, meaning that all field variables are assumed to be shared between the phases. To track the interface, an advection equation for the indicator function is solved. In order to obtain a sharp interface the discretization of the indicator function equation is crucial. (Stenmark et al., 2013)

2.8 Software

For geometry and mesh generation the ANSYS software ICEM CFD was used.

2.8.1 ICEM CFD

ICEM CFD is meshing software. It allows for the use of CAD geometries or to build the geometry using a number of geometry tools. In ICEM CFD a block-structured meshing approach is employed, allowing for hexahedral meshes also in rather complex geometries. Both structured and unstructured meshes can be created using ICEM CFD. (Stenmark et al., 2013)

2.8.2 Fluent Software

The Fluent solver is based on the centre node FVM discretization technique and offers both segregated and coupled solution methods. Three Euler-Euler multiphase models are available; the Eulerian model, the mixture model and the VOF model. In addition, one particle tracking model is available. As mentioned in Section 2.6.1, the discretisation of the volume fraction equation is crucial in a VOF method to keep the interface sharp. The choice of discretization method can have a great influence on the results in other multiphase models as well. To resolve this issue, Fluent has a number of discretisation techniques implemented specifically for the volume fraction equation. Several methods are also available for spatial discretisation of the other transport equations.

To model interphase transfer there is both a number of drag models available along with other transfer mechanisms such as lift forces and turbulent dispersion Fluent offers three main approaches to model dispersed phases with a two-fluid formulation. With the default settings it is assumed that the dispersed phase has a constant diameter or a diameter defined by a user-defined function. With this setting, phenomena such as coalescence and breakage are not considered. (Stenmark et al., 2013)

2.8.3 Computational domain and Mesh

A sketch of the geometry (a) and numerical grid for computational domain (b) investigated in this study. GAMBIT 2.4 mesh-generator is employed to perform all geometry generation and meshing. The whole computational domain is a rectangle with a length of 20 m and a height of 15 m. The length of computational domain is large enough, which is larger than the longest horizontal distance the oil droplets migrate when they reach the sea surface. Water occupies the lower region with height of 14.5 m, while air occupies the upper region. In the computational domain, the damaged submarine pipe with the outer diameter (D) of 0.6 m on both sides, the most common diameter of submarine pipe used in Bohai oilfield, is located in the sea bed. The size of the leakage hole (d) is a variable ranging from 0.1 m to 0.3 m with increment of 0.1 m, in order to examine the effect of leak size (H. Zhu et al., 2014)

Table 2.5 Simulation cases, in which variables of oil density, oil leaking rate, diameter of leak, and maximum water velocity are varied (*H. Zhu et al., 2014*)

Case	Oil density (kg/m^3)	The maximum water velocity(m/s)	Oil leaking rate (m/s)	Diameter of leak(m)	Volume flux of leaking oil(m^3/s)	Flux multiple (comparing with case 12)
1	780	0.1	2	0.05	0.003925	25
2	810	0.1	2	0.05	0.003925	25
3	840	0.1	2	0.05	0.003925	25
4	870	0.1	2	0.05	0.003925	25
5	900	0.1	2	0.05	0.003925	25
6	930	0.1	2	0.05	0.003925	25
7	960	0.1	2	0.05	0.003925	25
8	870	0.1	1	0.05	0.0019625	12.5
9	870	0.1	3	0.05	0.0058875	37.5
10	870	0.1	4	0.05	0.00785	50
11	870	0.1	5	0.01	0.0098125	62.5
12	870	0.1	2	0.02	0.000157	1
13	870	0.1	2	0.03	0.000628	4
14	870	0.1	2	0.04	0.001413	9
15	870	0.1	2	0.05	0.002512	16
16	870	0.04	2	0.05	0.003925	25
17	870	0.07	2	0.05	0.003925	25

3.0 MATERIALS AND MEHTODS

3.1 Overview

This paper is about to study the oil flows from damaged submarine pipelines with different water velocities. First and foremost, CFD (computational fluid dynamic) model coupling with VOF (volume of fluid) method has been used to investigate the process of oil spill from submarine pipeline to free surface. The actual shear velocity distribution of current and the actual hydrostatic pressure distribution are considered in this study. Detailed oil droplet and sea-surface in-formation could be obtained by the VOF model. Effects of leak size on the oil spill process are examined.

3.2 Simulation Methodology

3.2.1Governing equations

The VOF approach is based on the solution of one momentum equation for the mixture of the phases, and one equation for the volume fraction of fluid. In this study, volume of fluid functions F_w and F_o are introduced to define the water region and the oil region, respectively. The physical meaning of the F function is the fractional volume of a cell occupied by the liquid phase. For example, a unit vale of F_w corresponds to a cell full of water, while a zero value indicates that the cell contains no water. The fraction functions F_w and F_o are described as follows:

$$F_w=\frac{V_w}{V_c} \;\ldots\ldots\ldots\ldots\ldots\ldots \text{(eq. 3.1)}$$

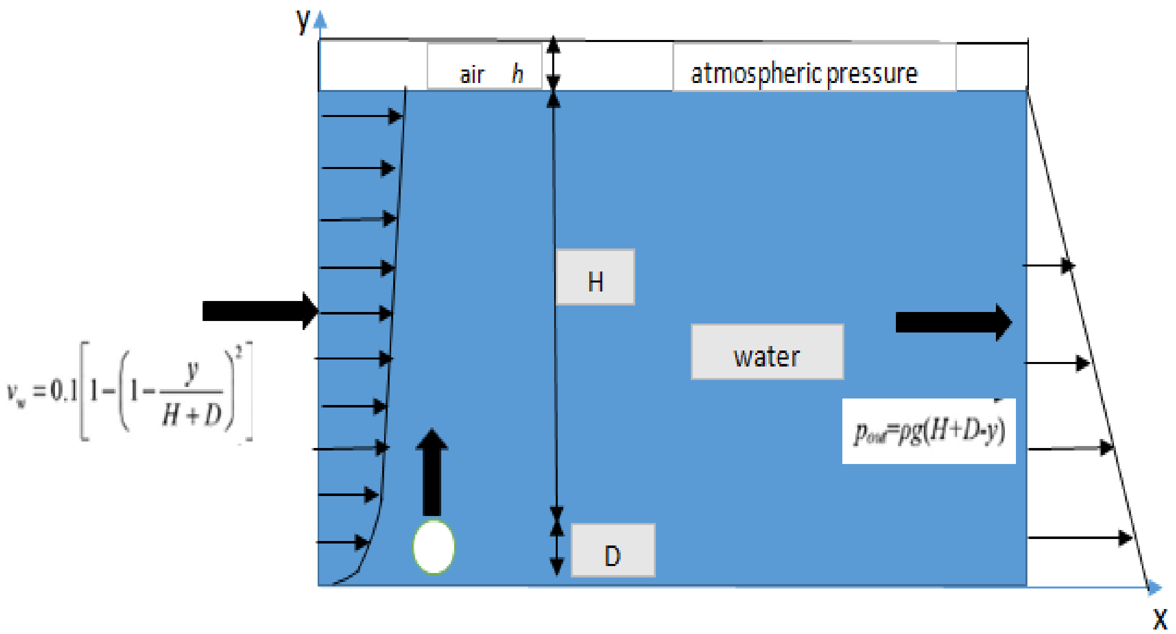
$$F_o=\frac{V_o}{V_c} \;\ldots\ldots\ldots\ldots\ldots\ldots \text{(eq. 3.2)}$$

Where F_o and F_w are oil and water fractional function, respectively, V_c, V_o and V_w represent volume of a cell, volume of oil inside the cell and volume of water inside the cell, respectively.

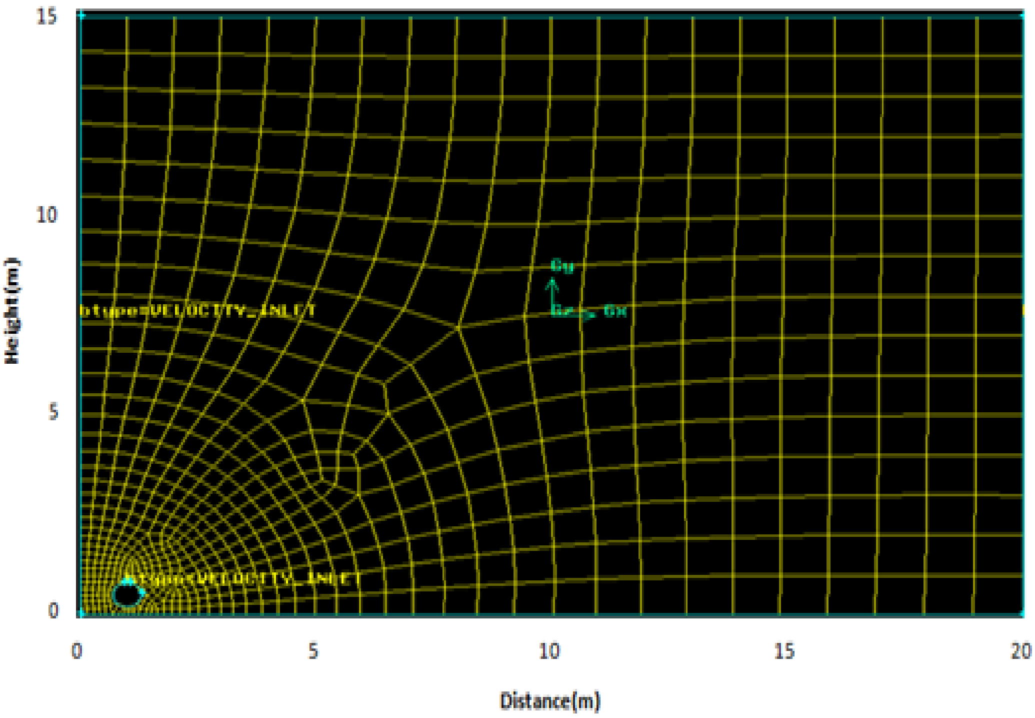
3.2.2 Computational domain and mesh

(a) Gambit 2.4.6

A two-dimensional flow simulation is accurate enough to capture the maximum horizontal migration distance. Using Gambit 2.4, a rectangular computational domain was created with a length of 20 m and height of 15 m. Water occupies the lower region with height of 14.5 m, while air occupies the upper region. In the computational domain, the damaged submarine pipe with the outer diameter (D) of 0.6 m was displayed and transformed to a coordinate of (-9,-7, 0). There is a leakage hole on the top of pipe, opening upwards. The size of the leakage hole (d) is set at 0.1m. Then, a paved quadratic mesh with 0.5 spacing was generated for the leak and domain at one time. Progressive mesh is used to capture the near-leak flow properties. A suitable grid density is reached by repeating computations until a satisfactory independent grid is found. The Gambit file is draw again for the different leak size 0.2m and 0.3m.The mesh is then exported to be used to generate results in the Fluent 6.3.26. Figure 3.1 below shows (a) sketch of the geometry and (b) numerical grid for computational domain investigated in this study.



(a)



(b)

Figure 3.2.2 Sketch of the geometry and numerical grid for computational domain: (a) overall view of the computational domain and boundary conditions; (b) grid distribution of computational domain. *(Zhu et al. 2014)*

(b) Fluent 6.3.26

The mesh document is then read in the fluent software with a 2d, dp unsteady simulator**.** Then, grid was checked and a multiphase theory was approached. Turbulence of k-epsilon was chosen to test its viscosity of kerosene liquid. The next procedure was to select materials in the multiphase category. The three phase materials chosen were water liquid, kerosene liquid and air at their fixed densities respectively.

The parameter of inlet water velocity (vs.) was fixed in this case. Phase 2 which consists of kerosene liquid was set with a volume fraction of 100%. Under the operating conditions, gravity of 9.81ms^{-2} was chosen acting in the vertical direction. The grid was displayed and initialization was begun.

To generate results, the animation was run based on time step. Sequence was chosen and the contour was set to phases. All ranges were ticked to enable visual interpretation of results. Finally, case input of data was checked and iteration step was chosen. A time step size of 0.1sec with 1000 number of time iterations was set. The simulation was run at 20 max iterations per time step.

3.3Effects of variables on the Oil Spill Process

3.3.1 Effect of leak size

The results indicate that the effect of the diameter of leakage hole plays a significant role in the spread of oil spill. With increasing leakage size, the time required for oil to reach the maximum horizontal migrate distance when it reaches the free surface is shortened. As leakage size reduces from 0.05m to 0.01m, the required time decreased by 23.53 percent. It can be explained that at the same leaking rate, the bigger the diameter of leak, the larger the amount of released oil and the greater the upward momentum. Due to the large mass flow rate, oil droplets released from the leak with d = 0.05 mare easier to collision and have greater chance of gathering into large droplets. Though the water velocities are the same, large active faces of big oil droplets lead to great shear stress. Under the action of shear stress, the maximum horizontal migrate distance,16.7m (18.5m minus 1.8 m), presents in the case of d=0.05 m. This distance is about 1.5 times than the maximum horizontal migrate distance for d =0.01m. Therefore, big-hole leaks may lead to more serious consequences.

Table 3.1 List of parameter used in this simulation

Parameters	**Visual**	**Multi phase**	**Viscosity**	**Materials**	**Phases**	**Oil leak rate (m/s)**	**Density (kg/m^3)**
Conditions	2d, dp	VOF	k-epsilon	-Water liquid -Air -Kerosene Liquid	1)Water liquid 2)Kerosene liquid	0.1	780 kg/m^3